

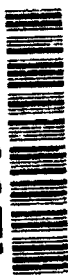
FILE 956 002

(2)

TECHNICAL REPORT RD-WS-92-13

DTIC
S **ELECTE** **D**
APR 22 1993
C

AD-A263 245



**PROTOTYPE AND SHORT-RUN
PRINTED CIRCUIT BOARD CREATION**

Darryl Hölder
Weapons Sciences Directorate
Research, Development, and Engineering Center

March 1993



U.S. ARMY MISSILE COMMAND

Redstone Arsenal, Alabama 35898-5000

Approved for public release; distribution is unlimited.

88 4 21 059

93-08611

DESTRUCTION NOTICE

FOR CLASSIFIED DOCUMENTS, FOLLOW THE PROCEDURES IN DoD 5200.22-M, INDUSTRIAL SECURITY MANUAL, SECTION II-19 OR DoD 5200.1-R, INFORMATION SECURITY PROGRAM REGULATION, CHAPTER IX. FOR UNCLASSIFIED, LIMITED DOCUMENTS, DESTROY BY ANY METHOD THAT WILL PREVENT DISCLOSURE OF CONTENTS OR RECONSTRUCTION OF THE DOCUMENT.

DISCLAIMER

THE FINDINGS IN THIS REPORT ARE NOT TO BE CONSTRUED AS AN OFFICIAL DEPARTMENT OF THE ARMY POSITION UNLESS SO DESIGNATED BY OTHER AUTHORIZED DOCUMENTS.

TRADE NAMES

USE OF TRADE NAMES OR MANUFACTURERS IN THIS REPORT DOES NOT CONSTITUTE AN OFFICIAL ENDORSEMENT OR APPROVAL OF THE USE OF SUCH COMMERCIAL HARDWARE OR SOFTWARE.

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188
Exp. Date: Jun 30, 1986

1a. REPORT SECURITY CLASSIFICATION UNCLASSIFIED			1b. RESTRICTIVE MARKINGS	
2a. SECURITY CLASSIFICATION AUTHORITY			3. DISTRIBUTION/AVAILABILITY OF REPORT Approved for public release; distribution is unlimited.	
2b. DECLASSIFICATION/DOWNGRADING SCHEDULE			5. MONITORING ORGANIZATION REPORT NUMBER(S)	
4. PERFORMING ORGANIZATION REPORT NUMBER(S) Technical Report RD-WS-92-13			7a. NAME OF MONITORING ORGANIZATION	
6a. NAME OF PERFORMING ORGANIZATION Weapons Sciences Directorate RD&E Center		6b. OFFICE SYMBOL (if applicable) AMSMI-RD-WS-UB	7b. ADDRESS (City, State, and ZIP Code)	
6c. ADDRESS (City, State, and ZIP Code) Commander, U.S. Army Missile Command ATTN: AMSMI-RD-WS-UB Redstone Arsenal, AL 35898-5248			9. PROCUREMENT INSTRUMENT IDENTIFICATION NUMBER	
8a. NAME OF FUNDING/SPONSORING ORGANIZATION		8b. OFFICE SYMBOL (if applicable)	10. SOURCE OF FUNDING NUMBERS	
8c. ADDRESS (City, State, and ZIP Code)			PROGRAM ELEMENT NO.	PROJECT NO.
			TASK NO.	WORK UNIT ACCESSION NO.
11. TITLE (Include Security Classification) Prototype and Short-Run Printed Circuit Board Creation				
12. PERSONAL AUTHOR(S) Darryl Hölder				
13a. TYPE OF REPORT Final		13b. TIME COVERED FROM Sep 90 TO Sep 92	14. DATE OF REPORT (Year, Month, Day) March 1993	15. PAGE COUNT 23
16. SUPPLEMENTARY NOTATION				
17. COSATI CODES			18. SUBJECT TERMS (Continue on reverse if necessary and identify by block number)	
FIELD	GROUP	SUB-GROUP	Printed circuit boards; transparencies; etching; personal computers	
19. ABSTRACT (Continue on reverse if necessary and identify by block number)				
<p>This report describes the knowledge and experience developed within this directorate over the past several months in the fabrication of Printed Circuit Board (PCB) techniques. Due to the limited quantity of identical, custom-made PCB's needed for use in the laboratory, it is both cost-effective and less time-consuming to fabricate PCB's in-house, than to employ an outside supplier. The combination of equipment and techniques permits the fabrication of single and short-run PCB's of up to 8 by 10 inches, with little problem. The system described here has been in place for about two years after being developed over a longer period. As with any technique-dependent process, experience and learning will contribute to better quality in the product. However, this set of equipment and the techniques described should yield reasonable results very quickly after the start of operation. Continuing improvement of these techniques is taking place, but there now exists a capability for the creation of small quantities of PCB's sufficient for laboratory uses. The design techniques used are directly useful for the creation of materials required by an outside jobber for producing large quantities of PCB's, should that be required.</p>				
20. DISTRIBUTION/AVAILABILITY OF ABSTRACT <input type="checkbox"/> UNCLASSIFIED/UNLIMITED <input checked="" type="checkbox"/> SAME AS RPT. <input type="checkbox"/> DTIC USERS			21. ABSTRACT SECURITY CLASSIFICATION UNCLASSIFIED	
22a. NAME OF RESPONSIBLE INDIVIDUAL Darryl Hölder			22b. TELEPHONE (Include Area Code) (205) 876-5177	22c. OFFICE SYMBOL AMSMI-RD-WS-UB

DD FORM 1473, 84 MAR

83 APR edition may be used until exhausted.
All other editions are obsolete.

SECURITY CLASSIFICATION OF THIS PAGE

UNCLASSIFIED

i/(ii Blank)

ACKNOWLEDGEMENTS

The author would like to thank Mr. R. Carnes of BDM International whose efforts contributed greatly to the development of the capability described here. Significant contribution toward the efficient use of design software was made by Mr. M. Faust, also of BDM International.

DTIC QUALITY INSPECTED 4

Accession For	
NTIS CRA&I	<input checked="checked" type="checkbox"/>
DTIC TAB	<input type="checkbox"/>
Unannounced	<input type="checkbox"/>
Justification	
By	
Distribution /	
Availability Codes	
Dist	Avail and/or Special
A-1	

TABLE OF CONTENTS

	Page
I. INTRODUCTION	1
II. CREATION OF PCB ARTWORK	1
III. PCB ARTWORK PREPROCESSING	3
IV. CREATION OF PCB TRANSPARENCIES	6
V. PCB ETCHING AND POST-PROCESSING	7
VI. CONCLUSION AND FINAL THOUGHTS	10
APPENDIX	A-1

I. INTRODUCTION

In research and development laboratories, there are many instances where custom electronic circuits are required. Quantities of these are rarely more than a dozen identical circuits, so there are no economies to be gained by employing an outside supplier to fabricate any electronic Printed Circuit Boards (PCBs) needed. The advantages of using a PCB are many. Just to name a few: there is unit-to-unit stability (such things as wideband op-amps must have tightly controlled stray impedances); fabrication is simplified (populating and soldering a properly designed PCB is much less prone to error than hand wiring); the reproduction time is reduced; and mechanical stability of the circuit is naturally provided by the PCB material.

This report is published for the purpose of documenting and propagating the knowledge of PCB fabrication techniques that has been developed within this directorate over the past several months of effort. Continuing improvement of these techniques is taking place, but there now exists a capability for the creation of PCBs of one-off or small quantities of quality sufficient for laboratory uses. The design techniques used are directly useful for the creation of materials required by an outside jobber for producing large quantities of PCBs, should that be required.

The efforts reported here were developed by many persons, government and contractor, both drawing information from many sources.

II. CREATION OF PCB ARTWORK

The first step in the creation of a PCB is to design the artwork. For this, the designer should have a schematic of the circuit to be created, the PCB size/shape requirements, and the physical characteristics of the components to be used. Connectors, switches, transformers, and even capacitors vary widely in their physical configurations, so these must be selected and the connection configuration defined, either from published dimensions or by actual measurement from the part to be used. Integrated Circuits (ICs), transistors, resistors, and such are much more standardized except for the choice of through-hole (standard) or Surface-Mount Technology (SMT) devices. Use of the SMT devices offers smaller size and less drilling of holes in the PCB, but since many SMT devices have fine-pitch leads, much greater care in the PCB etching and soldering operations is demanded. Only after substantial perfection of the techniques described below should fine-structure designs be attempted.

Computer aid in the creation of the basic PCB artwork is practically a must. Several PCB design packages are available for the popular desktop computers. For the DOS-based Personal Computers (PCs), Schema PCB and PADS PCB are used widely. For the Macintosh, McCAD-PCB is available. Referenced commercial products and vendors are listed in the Appendix. Using the software package suitable for the available computer, the layout of the components and the connecting circuit traces is created. For the methods of PCB creation used here, plated-through holes cannot be generated. Care must be taken to produce a design which will require soldering only on the side opposite that of the component if it is the usual through-hole type (not SMT). This places a restriction on circuit trace placement but prevents cramped and usually unsatisfactory attempts at soldering between a component and the top PCB surface. Some success has been obtained using miniature copper eyelets for imitation of plated-through holes. The combination of 0.039 inch diameter shank eyelets and 0.080 inch diameter pads can produce an acceptable feed-through. Massive application of this technique (as for all pins of a several-IC board) should be avoided since this would produce a very tedious and error-prone operation at

assembly time. With due care taken in matching the images at PCB exposure time, traces on both top and bottom of the PCB can be used with power and signals transferred between sides of the board by means of vias fitted with short pieces of bus-wire soldered on top and bottom.

It is critical that the artwork be thoroughly checked before further processing. It is all too easy to plot a mirror image of the pinout of an IC or other function module. Natural confusion can result from trying to visualize a component's footprint from both the top and bottom of the PCB; for that reason, most PCB software shows the top (component side) layer and bottom (solder side) in an overlaid X-ray view. With this being the case, test prints should be made on paper and overlaid on a back-lit translucent surface (light box). Utilizing a tabletop vue-graph displayer which is available from office supply stores at a reasonable cost gives acceptable light box performance. It almost goes without saying that at least two persons should check the design. It is usually best that the creator of the PCB artwork not be the designer of the circuit. The artwork designer can hardly avoid checking the design with the same mind-set with which the design was drawn, making it difficult to detect errors. By like sign, the circuit designer must be meticulous in the pin-number-by-pin-number checking so that errors such as running a pair of logic signals to inputs of separate gates rather than to the same gate may be avoided.

III. PCB ARTWORK PREPROCESSING

There are two types of photo-resist coatings available, positive and negative. The positive-resist type requires a photo master of the artwork to have the circuit traces, ground planes, and other areas where copper is to remain, to be opaque, and areas that are to be etched away to be transparent. The negative-resist type is just the opposite, that is, the dark areas get etched away. Direct printing of the layout file on most PCB design programs results in a positive image as above. Unfortunately, the most readily available and least expensive precoated photosensitive PCB stock is of the negative-resist type. This makes inversion of the image necessary (positive-to-negative and negative-to-positive transitions).

In the case of the designer using the Macintosh computer, an inversion can be readily made using the **Page Setup...** selection under the **File** menu. When the dialog box **Laserwriter Page Setup** appears, select **Options** (for a second dialog box). When this dialog box opens click in the check-box next to **Invert Image** to select it. A picture representing the document to the left of the check-boxes will change to indicate the option selected. While in these two dialog boxes, make other settings as appropriate to the master (such as image reversal - **Flip Horizontal** in this second dialog box, or scaling - **Reduce or Enlarge** in the first dialog box). After these settings are made, exit the **Page Setup...** selection and select **Print...** from the **File** menu. The resultant page output from the laser printer should be a negative (copper areas light, etched-away areas dark) and of the proper size.

For designers using PC platforms, the inversion is substantially more complicated. Inversion of an image requires that the header to the PostScript file be edited to cause the creation of an all-black background and the drawing of lines, figures, and areas to be made with the color white. Most applications that create PostScript files, regardless of the computer platform or the application type, will allow the saving of the PostScript file to disc. This file may be printed out, read, and edited using a text editor or word processor that will save ASCII text files. In this office, the PC-based PCB layout program is Schema PCB which uses the same PostScript printer driver as the PADS PCB layout program. The PostScript printer driver for our system shows that it was created by Andy Montalvo on 8 June 1990. The header portion is exactly 453 lines long (about 9.5 pages at 52 printed listing lines per page) and ends with the PostScript commands:

```
1 1 div dup scale
75 sg
50 sg
25 sg
0 sg
```

The next line will vary from one file to the next since this is the start of commands for drawing the data of a specific image; a typical line might be "20 20 10 SetPoly", etc.

To invert a Schema PCB or PADS PCB design file, load it into an editor program, make the following changes, save it, and use the PCB software as normally done to print it. Obtain a listing of the file printed using 52 printed listing lines per page:

Line #344 (page 7) – for the drawing of oval pads:

Original ... 1 setgray

Revised ... 0 setgray

Comment .. This will make setgray black so that the oval pad holes will show.

Line #359 (page 7) – for the drawing of thermal relief pads:

Original ... 1 setgray

Revised ... 0 setgray

Comment .. This will make setgray black so that the thermal pad holes will show.

Insert below Line #435 (page 9) – line reads “1.5 setmiterlimit 1 RndAper”

% BLACKFLASH ROUTINE – PROVIDES BLACK BACKGROUND

% This routine adapted from the PostScript roundtable (PSRT) on the

% GENie online information service hosted by Don Landcaster.

newpath

0 setgray

% Select Black

00 moveto

% Making a rectangle full page size:

8000 0 rlineto

% Move 8 inches across...

0 10000 rlineto

% and 10 inches up...

-8000 0 rlineto

% and 8 inches back ...

closepath

% and close figure to first point.

fill

% Make printable area all Black.

% END OF BLACKFLASH ROUTINE.

Line #436 (page 9) – for setting the line defaults:

Original ... 0 setgray

Revised ... 100 setgray

Comment .. This will make setgray white to draw lines using “white ink”.

Line #437 (page 9) – for setting the color defaults:

Original ... /glev 0 def

Revised ... /glev 100 def

Comment .. This will make “white ink” the default for drawing.

Lines #450 through #453 (page 9) – for presetting the setgray stack:

Original ... 75 sg

50 sg

25 sg

0 sg

Revised ... 25sg

50 sg

75 sg

100 sg

Comment .. This revises the setgray stack for the new gray levels needed to make the image print correctly.

Once all these changes are made, save the file and send it to the laser printer using the normal printing routines. If all has been correctly entered, an inverted image should be printed. To ease the editing drudgery, a fixed file of just the modified header could be saved which might be substituted in total for the usual header of the file generated by the application. A complicating

factor is that Line #443 (page 9) is a tag label unique to each project and would show whatever is enclosed within the parentheses of the modified header unless it is manually edited each time.

One additional printing item to be addressed is that of image reversal (mirror image transformation). In order to properly expose the photosensitized PCB stock, the toner side of the final transparency must rest directly on the photoresist layer; there must be no intervening layer of transparent film. This procedure prevents light-leakage by diffraction effects at edges of the image from greatly affecting the definition of the projected image into the photoresist layer. As a result, trace edges and fine structures will be sharply defined. The normal printing procedure places the toner on the top of the film which, when this side is pressed against the photoresist layer, will be proper for the bottom (solder) side of a PCB. In order to properly place the toner image for the PCB top (component) side, the image must be reversed at printing. This will place the toner on the film such that when the film is turned over and placed on the photoresist layer, the image is correct. Many PCB design packages will do this upon command. In case of doubt, produce test prints on paper, place them face-to-face, and view the resultant overlay against a strong light. The printing is correct when, with the two toner layers meshed, the PCB traces can be brought into alignment as was designed on the PCB layout software.

IV. CREATION OF PCB TRANSPARENCIES

Once the problem of positive or negative images is solved, the next concern is how to get a print on a transparency which is dense enough to yield proper exposure of the sensitized circuit board material. This brings up the subjects of transparency creation and densifying. The creation of a transparency is relatively simple using standard materials designed for overhead transparencies. This material is produced by several vendors and is available at most office supply stores as "transparency film for plain paper copiers." The film most used here is Arkwright PPC-IX although several other manufacturers supply a similar product. One major precaution is that a transparency film rated for the high temperatures involved in laser printers and xerographic reproduction be used – the usual "acetate" sheets used for hand-drawn overhead presentations are not suitable since they will melt and jam (and possibly damage) the fuser in the laser printer! When the PCB image file is ready, one simply prints that file to the laser printer using the one-sheet-at-a-time handfeed option to allow substitution of the transparency film rather than the normal use of paper from the laser printer's tray. Careful attention to the state of the laser printer cartridge and setting of the printer's density control can be used to produce a dense image, clean clear areas, and obtain good resolution of fine details.

Once the appropriate image is produced on the transparency film, it still must undergo a further treatment called "densifying". Even in the most dense laser printer image, the blacked-out areas do not block enough light to produce a proper exposure on the photosensitive PCB material. This can be readily verified by back-lighting the film with a strong lamp and inspecting those areas using a 4X to 9X magnifier; many pinholes will be seen. The process of densifying involves the application of a metallic transfer film to the toner areas on the transparency. This is done by means of a heat-and-pressure applique fuser. One such fuser is the Canon Color Fuser product # CC100. A suitable transfer film is OmniCrom Color Transfer Sheet – Gold Metallic thermal laminate. The process involves cutting a piece of the laminating material of sufficient size to cover the toner image and fixing it in place to the transparency film using small paper labels (plastic tape will also melt in the fuser), then feeding this assembly through the heated fuser. Take care to feed the sheets straight and flat because there is a tendency for the transfer sheet to wrinkle, which produces voids in the deposited metallic coating. Once the spent transfer sheet is carefully stripped from the PCB image film, there should remain a smooth, shiny coating of the transfer material over all of the toner areas. The image should be inspected with a light and magnifier as discussed above and any remaining small voids filled with a commercial PCB "resist pen" as sold in most electronics stores. It is important that these voids be filled on the film side opposite the toner/laminate so that these are not damaged by the pen point.

V. PCB ETCHING AND POST-PROCESSING

Operations included in the catch-all term "etching" include exposure and development of the photosensitized PCB resist coating and the actual chemical removal of the unwanted copper. Post-processing includes PCB size trimming, tin-coating, drilling, and other processes, the final result of which produces a PCB ready for electronic assembly.

The "etching room" should be an isolated, well-ventilated, enclosed area which can be made dark. One particular precaution is to exclude "pencil-beams" of light (as from pipe penetrations or door-jambs). A red photographic low-wattage safelight can be used during the photosensitive stages of operation. It is also handy to have a portable safelight to inspect the development of the PCB resist. This portable safelight can be either commercial or can be made from a red LED panel lamp and a battery holder.

The chemicals involved do present some physical hazard and as such must be stored in chemical lockers when not in use. The containers used must be of a sturdy, tightly-sealing, and light excluding design. When in use the operators must protect themselves by means of a chemical apron, rubber gloves, and eye protection. A full face-shield is recommended since it not only provides better protection, but is less prone to fogging. The chemicals used here are sodium-hydroxide based and as such, any items that come in contact with them must be non-metallic. Suitably-sized rectangular Pyrex cookware dishes are durable and inexpensive, and plastic tongs, funnels, etc. are available through chemistry suppliers.

The photosensitive PCB material should also be afforded proper storage. This is normally supplied in light-tight black plastic envelopes. As the sheets are used, any sizable trimmings can be returned to these envelopes for possible future use. A sturdy, lockable, cabinet with wide, shallow drawers is desirable since the PCB material should be stored flat to prevent warping. A mechanic's tool cabinet is a suitable inexpensive choice.

The equipment used here is a good choice for a lab where the demand for boards is sporadic. The primary units are the PCB shear, exposure frame, and spray-type etcher. The PCB shear is a Kepro model # MS-6 which has a capacity for board stock 0.093 inch thick by 12.125 inches wide. The exposure frame is a Kepro model # BTX-200A with an area of 12 inches by 12 inches. The etcher is a Kepro model # BTE-202 benchtop model which can handle boards up to 12 inches by 12 inches. Some convenient accessories are a chemical hot-plate and an unheated air blower. The chemical hot-plate is closely temperature-regulated and is used to gently warm water for some of the intermediate stages of development. The unheated air blower is used to gently dry the PCB between stages of treatment.

Once the photographic master is ready, the next operation is to make a contact print on the photosensitized PCB stock. A precaution to observe is to use fresh photosensitized PCB stock. Stock that is aged exhibits irregularities in its resist composition which lead to difficulties in proper etching. The result is that in order to get cleanly-etched edges, many times small details are lost.

A typical job involving a moderate-size double sided PCB design will now be described. All materials such as the photo-masters, chemicals, wash water, etc. are brought into the etching room and the door closed. There should be no reason to leave until the PCB is safely in the etcher. The etching fluid level in the etcher should be checked and topped up if needed. The etcher is then turned on to preheat at least 30 minutes before expected use. With good room lights on, the photo-masters for the front and back of the PCB are trimmed about one or two

inches larger than the largest outline. They are then placed face-to-face (toner surfaces adjacent), brought into precise registration, and carefully taped together along at least one edge. This produces a "pocket" in which a piece of PCB material may be placed, the tape holding the images in registration.

After the photo-masters are taped as above, the red safelight is turned on and the room lights are extinguished. When the room is so darkened, then a piece of photosensitive PCB material is selected from stock. If the piece is oversize, it may be carefully trimmed using the PCB shear - care must be taken to avoid damaging the photoresist coating by means of a soft cloth on the bed of the shear. Any usable PCB trimmings should now be stored and the cabinet closed. The board should be of about the largest outline size in width and long enough to protrude beyond the photo-masters in length, as mentioned above. The PCB material is carefully inserted into the "pocket" as described above and the films manipulated so that the slight bend produced by the board thickness is equal for both films to preserve the front-to-back image registration. When this is accomplished, the films and PCB material should be taped securely so that no slipping is possible. This assembly is now placed in the exposure frame for exposure of the first side.

The time needed in the exposure frame will vary for the equipment and materials used. For the Kepro BTX-200A exposure frame and G-C Electronics positive-resist PCB material used here, this time is six to seven minutes. This time is critical and can only be gauged by experience with a particular selection of equipment and materials. Underexposure will produce a board that cannot be etched because the resist will not be removed by the developing solution. Overexposure will cause thin spots in the resist that will cause trace cracking. After the first side is exposed, the PCB/photo-master assembly is inverted (taking care not to cause slipping of the photo-master) and exposure is made on the second side. After exposure is finished, the assembly is removed from the exposure frame and the PCB removed from the photo-master "pocket". Using care to prevent scratches and fingerprints, the photo-masters made as described here regularly produce six high-quality PCBs before replacement is required.

The PCB is now immersed in the developing solution which should be kept agitated by rocking the tray. After about 1.5 to 2 minutes, the PCB may be removed, washed in warm water, and inspected. If inspection shows that the resist has properly and completely dissolved from the appropriate places, then the process is done, otherwise it can be returned to the developing tray for an additional (shorter) time. When the resist has satisfactorily been developed, wash the PCB in cold water to fix the resist and clear away residual chemicals and debris. As before, it can be dried using unheated air from a blower. The room lights may now be turned on and the safelight extinguished. The developer chemical should now be returned to its storage bottle since it deteriorates upon exposure to air and light. When the PCB is dry, a final inspection under good lights with a magnifier should be made to insure that there are no breaks in the intended resist pattern. If there are breaks, they must now be repaired with a resist pen because etching is the next operation.

If the etcher had been filled and turned on at the start of the PCB processing as mentioned above, it probably will be fully up to operating temperature at this time. The etchant jets should be adjusted to provide a fine, even spray. The PCB is attached to the holding fixture and inserted into the etcher. After checking to assure that the PCB is being evenly covered by the etchant spray, the PCB is left to process for two minutes. After this first two minute etching, remove the PCB and inspect it closely under a good light (and magnifier if the PCB is complicated). If the PCB is not yet fully etched, reinsert it into the etcher and expose it to an additional 30-second etch. After this short etch, re-inspect the PCB for complete etching. This process of inspection

and 30-second etch may be repeated as many times as needed to get a fully etched PCB. Too short an etch time will leave copper residue still attached to the substrate and the probability that closely spaced separate features will remain shorted together. Over-etching, on the other hand, will not be as severe a problem if the resist pattern is well-formed. In this case circuit traces and pads will be a little smaller due to chemical undercutting at the edges of the resist pattern. The main hazard is the possible etching away of fine structures and circuit trace pinholes due to resist weak points. When the PCB has been etched to satisfaction, shut down the etcher, and rinse the PCB with cool water to remove traces of the etchant and any residue. Since the etchant chemical is reused for several jobs, the above etching times may be longer for older, more used, chemical. This is because of the diluting effect of copper residue from previous jobs.

After the PCB is fully dry, it may be cut to final size using the PCB shear. If any machining operations are to be done (such as special cutouts, holes, or edge shapes), they should be done at this time. After the PCB has had all machining operations done except through-hole drilling, it should be cleaned of all traces of resist and the copper thoroughly scoured using either a 3M ScotchBrite pad or a cloth with a water-paste of Ajax cleanser. When the PCB is fully clean, it should be immersed in a nonmetallic tray of tin-plate solution to provide a protective coating to the copper. The tinning solution should be agitated by rocking the tray, and the coating process monitored by observation. After sufficient tin-plate has been deposited, the PCB is removed, cleaned, dried with unheated air, and the chemical is stored away. The PCB, when dry, is ready for drilling and final assembly.

The drilling process, although conceptually simple, is quite a painstaking job and very tedious for a board of any complexity. A special drilling toolset must be maintained just for this purpose. This office uses the Dremel model # 395 high-speed drill and the Dremel model # 212 miniature drill press. For PCB through-hole drilling it is very important that sharp drills be used, that there be zero looseness in the drill press, and that the PCB be held tightly against a firm, flat back-up plate. To do otherwise will result in the circuit pads being torn off the PCB. In order to maintain sharp drills, it is necessary to use carbide bits made for PCB drilling. The usual PCB substrate (fiberglass) will dull a normal tool steel drill after only a few holes. The carbide drill bits are both expensive and very prone to breaking; this increases the imperatives that the PCB be held tightly against a flat, firm plate and that the drill press exhibit zero looseness. If eyelets are to be used on the PCB they are installed at this time. The normal eyelet application is to use a 0.039-inch diameter shank eyelet in a 0.040-inch diameter hole in the usual 0.062-inch thick circuit board material. Normally, a rolled flange eyelet and eyelet setting tool is used since this provides a firm clinching action and 360° contact. If, however, a multilead component must be blindmated to the PCB, the funnel type eyelet might be preferable since it would provide for easier insertion of the component leads. Once these operations are complete, the PCB is finished and ready for electronic final assembly.

VI. CONCLUSION AND FINAL THOUGHTS

The system described here has been in place for about two years after being developed over a longer period. Other techniques have been tried (specifically the direct deposition of the laser printer toner to bare copper-clad PCB material) and have been found wanting in one or more important factors. The referenced direct-toner method (which uses the laser printer toner image as the etch resist and dispenses totally with the photographic process) is very simple in concept, but the materials are inadequate at this time to produce PCBs of professional quality. Primary problems with this technique involve poor adhesion of the toner to the copper, and low density of the toner which makes it an inadequate blocker for the etchant chemical. In addition, this process would be all but impossible to use for a double-sided PCB because of image registration problems.

The combination of equipment and techniques discussed here permits the fabrication of single and short-run PCBs of up to 8 by 10 inches with little problem. With careful work at splicing the transparency film of the photo-master, PCBs can be made up to the size capacity (12 inches by 12 inches) of the etcher. As with any technique-dependent process, experience and learning will contribute to better quality in the product. However, the set of equipment and the techniques described should yield reasonable results very quickly after the start of operation.

APPENDIX

APPENDIX: Equipment, Materials, and Sources

PCB Layout and Printing

1. PCB Software, Macintosh: McCAD-PCB software for schematic layout, digital simulation, schematic capture and PCB design on the Macintosh series of computers.

Manufacturer: VAMP, Inc.
Product number: McCAD EDS-2 PCB-ST
Price \$1895.00

- Source for item 1:
VAMP Inc.
6753 Selma Avenue
Los Angeles, CA 90028
Phone: (213) 466-5533

2. PCB Software, PC-compatibles: Schema-PCB software for schematic layout, schematic capture and PCB design on PC-compatible computers.

Manufacturer: Ovation
Product number: Schema-PCB
Price \$1990.00

- Source for item 2:
Ovation
801 Presidential Drive
Richardson, TX 75081
Phone: (800) 553-9119

3. PCB Software, PC-compatibles: PADS-PCB software for PCB design on PC-compatible computers.

Manufacturer: PADS Software, Inc.
Product number: PADS-PCB
Price \$1500.00

- Source for item 3:
PADS Software, Inc.
119 Russell St., Ste # 6
Littleton, MA 01460
Phone: (508) 486-9521

4. Laser Printing: PostScript roundtable (PSRT) on the GENie online telecommunications service hosted by Don Landcaster or the Landcaster-operated PostScript hotline at his company, Synergetics.

Price ... If registered user of GENie, standard connect-time charges apply to use of PSRT. For the Synergetics PostScript hotline, long-distance telephone charges apply.

- Source for item 4:
Synergetics
3860 West First Street
Thatcher, AZ
Phone: (602) 428-4073

Photo-Master Creation

1. Laminating Fuser: For creating printed circuit etching masks.

Manufacturer: Canon
Product number: Color Fuser # CC 100
Price \$349.00

2. OmniCrom Color Transfer Sheets: Box of 100 sheets 8.5" x 11" size, gold-color metallic thermal laminating material.

Product number: N A
Price (per box) \$69.95

- Source for items 1 and 2:
Lazer Products
12741 E. Caley Avenue, Unit 10
Englewood, CO 80111
Phone: (303) 792-5277

3. Transparency Film: For plain paper copiers/laser printers, in boxes of 100 sheets.

Manufacturer: Arkwright (and others)
Product number: PPC-IX
Price (per box) \$20.00

- Source for item 3:
Office supply stores

Etching Room Equipment and Chemicals

1. PCR Shear: Hand operated bench-top shear for PCB material to 0.092" thickness and sheet metal up to 0.040" thickness. Convertible with accessories for punching, notching, drilling, routing, and press brake operations.

Manufacturer: Kepro
Product number: MS-6
Price \$445.00

2. PCB Exposure Frame: Bench-top unit for exposure of photosensitive PCB material and photo master to ultraviolet light, with 10-minute timer for automatic shutoff of lamps, foam padded pressure lid, and locating pin registry rails.

Manufacturer: Kepro
Product number: BTX-200A
Price \$585.00

3. PCB Spray Etcher: Bench-top unit for etching of developed PCB material. Double-sided spray system with 1000-Watt titanium heater and regulator to maintain four gallons of etching chemical to the efficient 105° F \pm 5° temperature. With timer for automatic shutoff of chemical pumps, and "see-through" etching vat lid with safety interlocks.

Manufacturer: Kepro
Product number: BTE-202
Price \$865.00

- Source for items 1, 2, and 3:
Kepro Circuit Systems
630 Axminister Drive
Fenton, MO 63026-2992
Phone: (800) 325-3878

4. Plastic Forceps: Glass-filled Nylon disposable forceps, 11 centimeters long.

Manufacturer: Cole-Parmer
Product number: YB 06443- 10
Price (pack of 100) \$40.00

5. Plastic Funnel: Polypropylene vented funnel, 67 millimeter top diameter.

Manufacturer: Cole-Parmer
Product number: YB 06124-20
Price (bag of 100) \$24.40

6. Chemical Hot Plate: Temperature-regulated hot plate with cast aluminum top 6.25 inches square. Stepless thermostat regulates temperature from 10° C above ambient to 370° C with accuracy of \pm 3° C or better. Heater 120 Volts, 50/60 Hz, 750 Watts.

Manufacturer: Cole-Parmer
Product number: J-3400-00
Price \$105.00

- Source for items 4, 5, and 6:
Cole-Parmer Instrument Company
7425 North Oak Park Ave.
Chicago, IL 60648
Phone: (800) 323-4340

7. Laboratory Gloves: Utility laboratory gloves, neoprene-natural latex, 11.5" long.

Manufacturer: Playtex
Product number: 979
Price (per dozen) \$15.72

8. Laboratory Face Shield: Head frame of glass-reinforced plastic with replaceable acetate wrap-around 8" x 19" x 0.040" thick shield.

Manufacturer: NA
Product number: 878
Price \$12.96

9. **Portable Eye Wash Station:** Bench-top/wall-mount unit holding 2.5 liters of water with quick-release hands-free operation. Provides washing time of 4 to 6 minutes.

Manufacturer: Nalgene
Product number: 1219
Price \$78.00

10. **Chemical Apron:** Rubber coated, reinforced heavy cloth apron. Cloth tie-tapes at waist and neck.

Manufacturer: NA
Product number: 926
Price \$5.00

- Source for items 7, 8, 9 and 10:

Lab Safety Supply Co.
P. O. Box 1368
Janesville, WI 53547
Phone: (800) 356-0783

11. **Positive PCB Stock:** Pre-Sensitized PCB material-Positive type, 0.0625" thick glass-epoxy base (G-10). With one ounce (0.0014" thick copper cladding and Positive-acting photosensitized resist coating on both sides. Size 12 inches x 12 inches.

Manufacturer: GC Electronics
Product number: 22-357
Price (each) \$16.32

12. **Resist Pen:** Black felt tip pen with ink formulated to be an effective PCB etch resist. For PCB etch resist pattern modifications or last minute repairs before etching.

Manufacturer: GC Electronics
Product number: 22-220 (1/32" tip), 22-222(1/64" tip)
Price (each) \$1.19

13. **Resist Lacquer:** Lacquer for PCB etch resist pattern modifications or last minute repairs before etching.

Manufacturer: GC Electronics
Product number: 22-244
Price (0.5 oz. bottle) \$1.76

14. **Developer:** Concentrate of chemical to develop positive type photosensitized PCB resist patterns. Eight ounces diluted three to one will develop 10 to 20 square feet of PCB surface.

Manufacturer: GC Electronics
Product number: 22-226
Price (6 oz. bottle) \$3.57

15. **Stripper:** Solution for stripping etch resist coating from PCB surface.

Manufacturer: GC Electronics
Product number: 22-240
Price (2 oz. bottle) \$1.11

16. Etching Solution: Solution of ferric chloride for the etching of PCB surface.

Manufacturer: GC Electronics

Product number: 22-239

Price (1 gallon) \$8.59

17. Tin-plate Solution: Electroless tin-plating solution for the plating of PCB copper areas with a protective coating of tin, in six-ounce bottles.

Manufacturer: GC Electronics

Product number: 22-228

Price (1 gallon) \$1.54

- Source for items 11, 12, 13, 14, 15, 16, and 17:

GC Electronics

1801 Morgan Street

Rockford, IL 61102

Phone: (815) 968-9661

Post-Processing Equipment

1. PCB Drill Motor: Variable speed (5,000 to 30,000 RPM in five steps) miniature electric drill with ball-bearing construction and keyless chuck for 1/8-inch shank miniature bits and grinding tools. In kit with an assortment of grinding, sanding, polishing, and cutoff tools, collet for 3/32-inch tools, maintenance tools, and case.

Manufacturer: Dremel

Product number: 3950 (above kit containing model 395 drill)

Price \$104.00

2. PCB Drill Press: Miniature drill press for Dremel electric drills models 395, 285, and 275. Bench-top unit with six-inch square work surface with guide/hold-down slots, three-inch throat depth, and tool carriage adjustable up to three inches.

Manufacturer: Dremel

Product number: 212

Price \$42.75

- Source for items 1 and 2:

Allied Electronics

401 East 8th Street

Fort Worth, Texas 76102

Phone: (800) 433-5700

3. PCB Drills: Solid carbide drills with 1/8-inch shank in selected drill sizes from # 31 to # 70 size. Product number is CD-nnn, where nnn is the drill size in 1/1000s of an inch, i.e., a # 31 drill measures 0.120 of an inch so the product number would be CD-120.

Manufacturer: Kepro
Product number: CD-nnn
Price Varies with size from \$9.00 for largest (CD-126) to
\$4.50 for smallest (CD-028).

- Source for item 3:
Kepro Circuit Systems
630 Axminister Drive
Fenton, MO 63026-2992
Phone: (800) 325-3878

4. PCB Eyelets: Miniature eyelets fabricated for the electronic PCB industry, also eyelet setting and PCB modification tools.

Manufacturers: PCB Eyelets, Inc. and International Eyelets, Inc.
Product number: N A
Price Varies with size and plating type, typically \$10.00
per 100, tin or nickel plated.

- Source # 1 for item 4:
PCB Eyelets, Inc.
P.O. Box 550
Goleta, CA 93116-0550
Phone: (805) 968-3362
- Source # 2 for item 4:
International Eyelets, Inc.
5674 El Camino Real, Ste. L
Carlsbad, CA 92008
Phone: (800) 333-9353

INITIAL DISTRIBUTION LIST

	<u>Copies</u>
U.S. Army Materiel System Analysis Activity ATTN: AMXSY-MP (Herbert Cohen) Aberdeen Proving Ground, MD 21005	1
IIT Research Institute ATTN: GACIAC 10 W. 35th Street Chicago, IL 60616	1
Director U.S. Army Research Office ATTN: SLCRO-PH P.O. Box 12211 Research Triangle Park, NC 27709-2211	1
Headquarters Department of the Army ATTN: DAMA-ARR Washington, DC 20310-0623	1
Headquarters OUSDR&E The Pentagon Washington, DC 20310-0623	1
AMSMI-RD	1
AMSMI-RD-CS-R	15
AMSMI-RD-CS-T	1
AMSMI-GC-IP, Mr. Fred H. Bush	1
AMSMI-RD-WS, W. Wharton	1
J. Bennett	1
S. Troglen	1
AMSMI-RD-WS-UB, D. Hölder	25
RTTC, STERT-TE-E-EM, James Knaur	1